American Journal of Engineering Research (AJER)
e-ISSN: 2320-0847 p-ISSN : 2320-0936
Volume-6, Issue-3, pp-86-89
www.ajer.org
Research Paper Open Access

A Concise Description of One Way and Two Way Coupling Methods for Fluid-Structure Interaction Problems

MdFayaz Ahamed[1], Sharmin Atique[2], MdAbulKhair Munshi[3], Tuomas Koiranen[4]

[1]*Department of Computational Engineering, Lappeenranta University of Technology, Finland*
[2]*Department of Basic Science, National Institute of Textile Engineering and Research, Bangladesh*
[3]*Department of Mathematics, National University, Bangladesh*
[4]*Department of Chemical Engineering, Lappeenranta University of Technology, Finland*

ABSTRACT:*The interaction between fluid and structure may appear in many problems of Engineering. These problems related differential equations are very difficult to solve analytically and hence solved generally by numerical approximations. There is, however, a computational difficulty always arises to yield a desired convergence due to convoluted geometries, complicated physics of the fluids, structural deformations, and complex fluid-structure interactions. This computational complexity can be reduced by setting the interactional behaviour between fluid and structure. In this paper, several approaches have been discussed in order to reduce computational effort with a desired accuracy.*

Keywords: *Fluid-structure interactions (FSI), monolithic, partitioned, one-way, and two-way coupling.*

I. INTRODUCTION

The interaction between fluid and solid is a phenomenon which occurs in a system where a solid structure may deform due to fluid flow surroundings or inside it. Therefore, this deformation changes the boundary condition and shape of a fluid system. This can also happen the other way around where the structure changes the fluid flow properties. These are called Fluid-structure interactions (FSI) problem. Fluid–structure interactions can be stable or oscillatory. In oscillatory interactions, the strain induced in the solid structure causes it to move when the stress is reduced, and the structure returns to its former state only for the process to repeat. A flying aircraft or a running car is an example of FSI. It appears in many natural phenomena and mechanical systems. FSI plays a very important role in the design and analysis of many engineering systems.

FSI problems can only be calculated using laws and equations from several physical disciplines. Examples like this, where another sub problem occurs cannot be solved independently, are called multiphysics applications. These problems are often very complex and governing equations expressing the physical phenomenon of FSI problems in general are nonlinear partial differential equations. It is very cumbersome then to have analytical solutions of such problems. Therefore, they have to be solved by means of numerical solutions or experiments.

FSI is a subset of multiphysics applications and is defined well by Zienkiewicz and Taylor [1]:

"Coupled systems and formulations are those applicable to multiple domains and dependent variables which usually describe different physical phenomena and in which neither domain can be solved while separated from the other and neither set of dependent variables can be explicitly eliminated at the differential equation level."

II. METHODS

In general, Fluid-Structure Interaction problems are multiphysics problems which are very difficult to solve by analytical approach. Therefore, they are to be analysed either by using numerical simulations or experiments. Advanced discretization methods and availability of modern softwares in fields of computational fluid dynamics (CFD) and computational structural dynamics (CSD) have made this numerical solution possible. COMSOL multiphysics is one of the user friendly softwares for a multi-physics problem. Some other popular softwares in this field are ANSYS, Abaqus, ADINA, and so on. There are mainly two approaches for solving FSI problems using these softwares namely monolithic approach, and the partitioned approach.

In monolithic approach, the governing equations of fluid flow and displacement of the structure are solved simultaneously by a single solver. In other words, monolithic solution method directly runs fluid and structure equations by a unified algorithm. The interfacial conditions of fluid and structure are implicit in this solution approach. This process is better and more accurate for a multi-physics problem, but it may demand more resources and computational memory to develop and maintain such a specialized algorithm [2]. The mesh is connected here by non-conforming mesh methods. The following figure 2.1 illustrates the flow process of this approach:

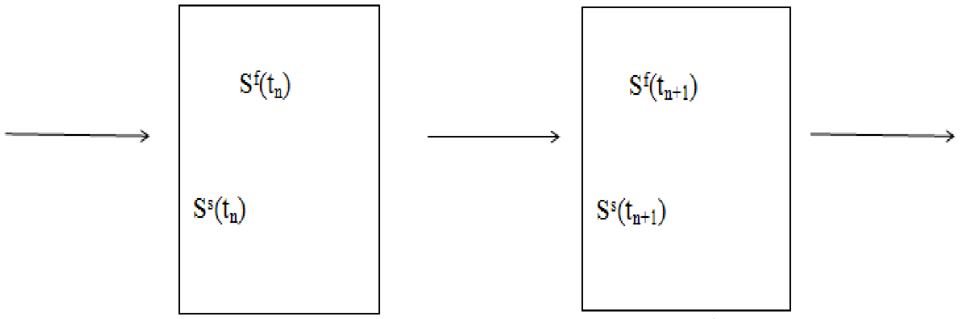

Figure 2.1: the monolithic approach for fluid-structure interactions, where S^f and S^s denote fluid and structural solutions respectively.

Partitioned approach is another way for solving FSI problems. Details about portioned method can be found in the literature [3-10]. In this method, sub programs are solved individually so that the structural solution does not change at the same time when fluid flow solution is computed. The governing equations of fluid flow and structural displacement are solved separately according to numerical solver and mesh discretization. Software modularity is preserved in partitioned approach as an existing flow and structural solver are coupled. Here, information is interchanged at the interface according to the coupling technique applied [11]. The interfacial conditions of fluid and structure are explicit in partitioned approach. A motivation of this approach is to combine fluidic and structural algorithm and to decrease the computational time. Hence, a partitioned method can be used to compute FSI problems with sophisticated fluid and structural physics [2]. A conforming mesh method is mainly used here to connect the mesh. The figure 2.2 below depicts the steps of a partitioned approach.

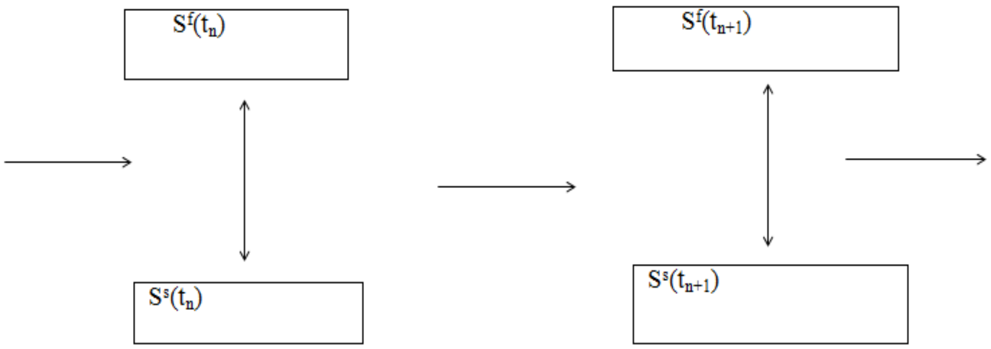

Figure 2.2: the partitioned approach for fluid-structure interactions, where S^f and S^s denote fluid and structural solutions respectively.

III. COUPLING

A coupling is said to be one-way if the motion of a fluid flow is affected by structural deformation and vice versa. Ship propeller is an example of this kind. In this coupling method, fluid flow is calculated first up to the desired convergence. After that resulted fluid flow calculation is interpolated to the structural model at the interface. Then, the structural model calculation is iterated until the desired convergence is achieved.

In a two-way coupling, fluid flow is affected by structural deformation and at the same time structural deformation is affected by fluid flow. Wind power point is an example of a two-way coupling. Here, converged solutions of the fluid flow influence the solid body deformation when the first time step runs. Then the result of fluid flow calculation is interpolated to the structural mesh at the interface as one-way coupling and the result of the structural solver is obtained from the fluid flow solution considering it as a boundary condition. Hence, the mesh of the structure is displaced and the displacement values are interpolated to the fluid flow mesh [12]. The process is iterated until the desired accuracy is achieved.

Regardless of whether one-way or two-way coupling methods are used, the solutions are based on a partitioned method where segregated solutions for the different physical fields are solved at the same time. One field that has to be prepared for fluid dynamics, the other is for structure dynamics. At the boundary between fluids and solids, the fluid-structure interface, information for the solution is exchanged between the fluid solver and structure solver. The information shared is based on the coupling method. For one-way coupling calculations, only the fluid pressure acting at the structure is transferred to the structure solver. For two-way-coupling calculations, the displacement of the structure is also transferred at the same time to the fluid solver [13].

In Figure 3.1 below, the solution procedure is shown for one-way coupling. At the beginning, the fluid field is solved until the convergence criteria are reached. The calculated forces at the structure boundaries are then transferred to the structure side. Then the structure side is calculated until the convergence criterion is reached. Then, the fluid flow for the next time step is calculated to convergence. The solution is finished when the maximum number of time steps is reached [13].

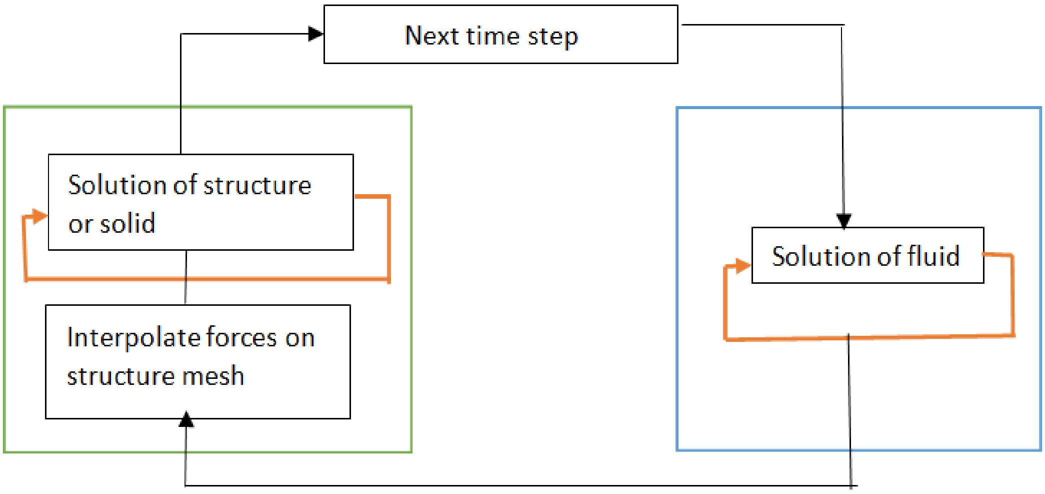

Figure 3.1: Steps of one-way coupling method

The steps of strong two-way coupling algorithm is shown on the following Figure 3.2. Inside one-time step during the transient simulation, a convergedsolution for the flow field is required to provide the forces acting on the body. A converged solution of the structural dynamics will be achieved under the response of the acting forcesonce interpolating the forces from the fluid mesh to the surface mesh of the structure. Theeffect of the structure to the inchoating load represents a displacement of the structural gridnodes. The displacements at the boundary between structure and fluid are interpolated to the fluid mesh which leads to its deformation. This step closes one inner loop of the simulation.For strong two-way-coupling simulations, these steps are repeated until the changes in theflow forces and the structural displacements fall below a certain amount. Next, anew time step is launched. For weak two-way coupling simulations, the convergence at theboundary between structure and fluid is not considered and a new time step is launcheddirectly [13].

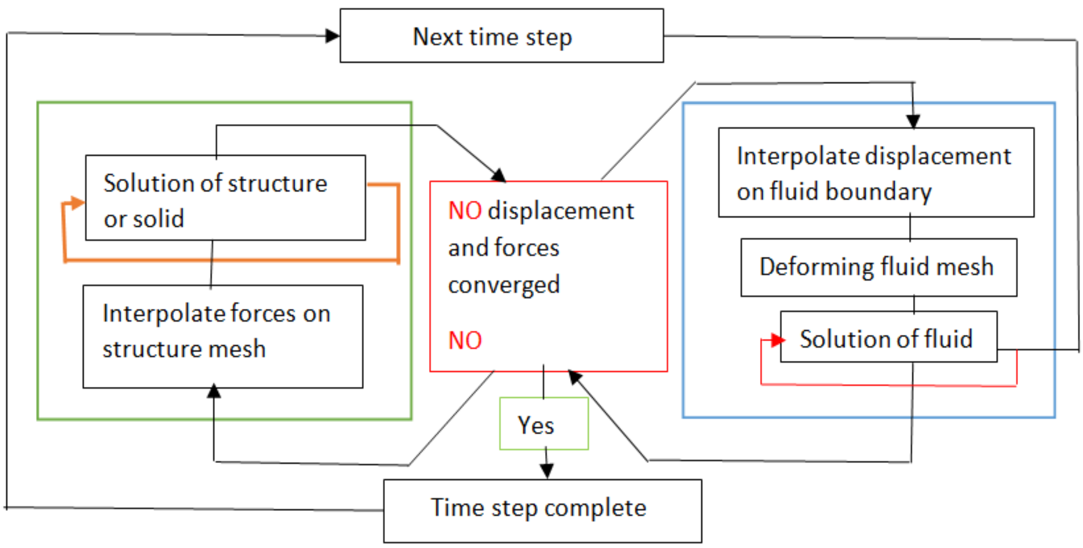

Figure 3.2: Steps of strong two-way coupling method

IV. DISCUSSION

The partial differential equations that govern fluid flow and define the solid nature are not usually amenable to analytical solutions, except for very simple cases. Therefore, in order to analyse fluid flows, flow domains are divided into smaller subdomains (made up of geometric primitives like hexahedra and tetrahedral in 3D and quadrilaterals and triangles in 2D). The governing equations are then discretized and solved inside each of these subdomains with proper boundary conditions. Meshing should be fine and appropriate to have a desired solution with minimum error. Hence, to ensure proper continuity of solution across the common boundaries between two subdomains they must be made meticulously so that the approximate solutions inside various portions can be put together to give a complete picture of fluid flow in the entire domain. The subdomains are often called elements or cells, and the collection of all elements or cells is called a mesh or grid. There are three discretization methods usually used to solve the system of partial differential equations, namely, finite volumes, finite elements, and finite differences.

V. CONCLUSION

The governing equations describing the physics of fluid-structure interaction problems are generally system of partial differential equations and hence having numerical solution for such multiphysics problem is often challenging.Failure to obtain full convergence is a common phenomenon for simulations carried out by using CFD softwares. Therefore, it requires a clear understanding in every steps of approximation process and perform a sensitivity analysis to avoid such failure. A proper apprehension in coupling methods may help to obtain a simulation that is both accurate and cost effective.There are some basic differences in calculation processes in both methods and therefore numerical solution also varies.

REFERENCES

[1]. Zienkiewicz and R. L. Taylor, “The Finite Element Method, vol. 1, Butterworth-Heinemann”, 5th edition, 2000.

[2]. Hou, G., Wang, J., & Layton, A., "Numerical Methods for Fluid-Structure Interaction — A Review", Commun. Compute. Phys., 2012, pp.337-377.

[3]. Sch¨afer, G. Sieber, R. Sieber, and I. Teschauer, “Coupled fluid-solid problems: examples and reliable numerical simulation,” in Proceedings of the Trend in Computational Structural Mechanics, International Center for Numerical Methods in Engineering _CIMNE_, Barcelona, Spain, May 2001.

[4]. K. C. Park, C. A. Felippa, and J. A. DeRuntz, “Stabilization of staggered solution procedures for fluid-structure interaction analysis,” Computational Methods for Fluid-Structure Interaction Problems, American Society of Mechanical Engineers, New York, NY, USA, 1977, T. Belytschko and T. L. Geers, Eds., vol. 26, pp. 95–124.

[5]. C. A. Felippa and K. C. Park, “Staggered transient analysis procedures for coupled mechanical systems: formulation,” Computer Methods in Applied Mechanics and Engineering, vol. 24, no. 1, 1980, pp. 61–111.

[6]. S. Piperno, C. Farhat, and B. Larrouturou, “Partitioned procedures for the transient solution of coupled aeroelastic problems, part I: model problem, theory and two-dimensional application,” Computer Methods in Applied Mechanics and Engineering, vol. 124, no. 1-2, , 1995, pp. 79–112.

[7]. S. Piperno, “Explicit/implicit fluid/structure staggered procedures with a structural predictor and fluid subcycling for 2D inviscid aeroelastic simulations,” International Journal for Numerical Methods inFluids, vol. 25, no. 10, 1997, pp. 1207–1226.

[8]. D. P. Mok, W. A. Wall, and E. Ramm, “Accelerated iterative substructuring schemes for instationary fluid structure interaction,” Computational Fluid and Solid Mechanics, K. J. Bathe, Ed.,Elsevier, 2001, pp. 1325–1328.

[9]. F. Nobile, Numerical approximation of fluid structure interaction problems with application to haemodynamics, EPFL, Switzerland, Doctoral thesis, 2001.

[10]. D. Peric and W. G. Dettmer, “A computational strategy for interaction of fluid flow with spatial structures,” in Proceedings of the 5th International Conference on Computational of Shell & Spatial Structures(IASS IACM’05), 2005.

[11]. Richter, T.," Numerical methods for fluid–structure interaction problems", Institute for Applied Mathematics, University of Heidelberg, Germany, 2010.

[12]. Raja, R. S., Coupled fluid structure interaction analysis on a cylinder exposed to ocean wave loading, CHALMERS UNIVERSITY OF TECHNOLOGY,Göteborg, Sweden ,Master’s Thesis in Solid and Fluid Mechanics ,2012.

[13]. F.-K. Benra,H. J. Dohmen, J. Pei,S. Schuster,and Bo Wan, “A Comparison of One-Way and Two-WayCoupling Methods forNumerical Analysis ofFluid-Structure Interactions” Journal of Applied Mathematics,vol.2011,2011.